ISSN 2063-5346

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF OPTIMIZED CONE NOZZLE DESIGN TO SPRAY UNIT OF AGRICULTURE UNMANNED AERIAL VEHICLE

Gowtham K[1]
Post Graduate Student, Department of Mechanical Engineering,
KONGU ENGINEERING COLLEGE, Erode-638060, Tamil Nadu, India

Dr. Hariharan V [2]
Professor, Department of Mechanical Engineering,
KONGU ENGINEERING COLLEGE, Erode-638060, Tamil Nadu, India

Article History: Received: 01.02.2023 **Revised:** 07.03.2023 **Accepted:** 10.04.2023

Abstract

Unmanned aerial vehicles has been gaining more attention in recent years in the agriculture field. Employment of the Unmanned Aerial Vehicles in agriculture field is to enable the operation related to crop monitoring and crop protection. Crop protection in agriculture field is enabled by employing the UAV spray as automated application. UAV spray carries fertilizer or pesticide. However, spray drift of the UAV is considered as major impact of the agriculture drones. In addition, various spray factors such as velocity, speed, height, orientatation of the nozzle play significant role in spray quality of the agriculture drone. In this paper, computational fluid dynamic model for designing optimal flow characteristic of spraydroplet at optimal velocity and orientation is designed and simulated in this work for effective spray deposit in the particular environment without losses. Design of Cone Straight Nozzle optimization on Nozzle Geometry using Controlled variable Method is carried out. Design considers the downwash properties of conestraight nozzles to the operation of dispersal and deposition of the pesticide or fertilizer. Further design of the nozzle of the spray unit in the drone is carried out on nozzle velocity combination using solid works to increase the quality of the spray and reduces the spray drift. Using ANSYS Workbench numerical simulation software, turbulence model is simulated on basis of the nozzle geometry for uniform distribution of the various chemical to the specified environment. Model is analyzed on various spray pressures, converging angle and throat length for uniform distributions of the spraying chemical. Finally,it is established on the various setting of discrete phase model at nozzle exit to improve the performance of the model to structure the spray deposit on basis of target and its environment. Result of the model is used to improve spray quality of the agriculture UAV in depositing the dose at right quality is measure in mass flow rate.

Keywords: Unmanned Aerial Vehicles, Cone Nozzle Design, Computation Fluid Dynamic, Discrete Phase Model, Spray Distributions, Nozzle Geometry

1. Introduction

Unmanned Aerial vehicle is gaining more advantage in the area of the agriculture as it largely reduces the human efforts and economic cost. Especially UAV is employed as fertilizer spraying application or pesticide spraying application towards plant protection [1]. In order to increase its utilization to application, various improvement has been made in existing literature with respect to movement trajectory of the UAV and deposition position of the UAV for pesticide droplet. However, it still lags in spray quality of downwash airflow nozzle of the rotor [2]. Further design of the nozzle of the spray unit in the drone is carried out on basis of the nozzle geometry for uniform distribution of the various chemical to the specified environment using solid works to increase the quality of the spray to the target and reduces the spray drift.

In order to improve the spray quality of the UAV for pesticide or fertilizer application, spary drift and other spray factors of the nozzle such as velocity, speed, height, orientatation, convergence angle and throat length of the nozzle has to be considered as it plays a significant role in spray quality of the agriculture drone to the particular target [3]. In this paper, computational fluid dynamic model for designing optimal flow characteristic of spray droplet at optimal velocity and orientation of nozzleto deposit the dose with respect to target is designed and simulated in this work. Design considers theCone Straight Nozzle optimization on Nozzle Geometry using Controlled variable Method for downwash properties of cone nozzles to the operation of dispersal and deposition of the pesticide or fertilizer [4].

Using ANSYS Workbench numerical simulation software [5], turbulence model [6] for downwash flow characteristics and velocity distribution with respect to velocity of the drone and spiral effects of cone nozzle are simulated. Finally, it is established on the various setting of discrete phase model at nozzle exit to improve the performance of the model in the specified trajectory of the drone for effective droplet distributions. Numerical Result of the model is used to improve spray quality of the proposed agriculture UAV design. Further performance analysis is carried out on cone straight nozzle and streamlined nozzle on flow characteristics.

Rest of the paper is organized into following section which is as follows, section 2 discusses the review of literature related to the proposed analysis. Section 3 provides the objective of the works and research methodology towards establishing the optimal flow characteristics and optimal velocity of the drone for spray application of the agriculture UAV with respect to nozzle design. Section 4 provides the simulation analysis of proposed research methodology using ANSYS software with respect to computational fluid dynamics. Finally, section 5 provides the conclusion of work

2. Related works

In this section, flow characteristics and velocity distribution to reduce the spray drift and various effect impacting the spray quality is analyzed using computation fluid dynamics models to the design of agriculture UAV. Further simulation analysis of those design and its properties is evaluated using the ANSYS software.

2.1. Computation Fluid Dynamic model using variation law for effective downwash airflow of various nozzle type of agriculture UAV for plant Protection

In this literature, effectiveness of the unmanned aerial vehicle employed for pesticide spray is analyzed and simulated to various flow characteristic and velocity distributions of the downwash airflow generated by the quadrotor at different flight velocities. Analysis represents that velocity exceeding the flight height of UAV will tend to cause droplet effect [7]. Further on employment of variation law, effective velocity distribution range and flow characteristics is identified. Spray characteristics depends on biological effectiveness and external conditions.

2.2. Effective Design and analysis of the computation of fluid dynamics of UAV

Downwash Air flow based on strain effect

In this literature, accurate measurement of the downwash flow trajectory and position of dropletof the unmanned aerial vehicle employed for pesticide spray is analyzed and simulated to various flow characteristic and velocity distributions which causes strain effects. Further optimization design to reduce the strain effects is modeled by solving the strain force with respect to the wind velocity. Finally, employment of optimization, effective velocity distribution range and flow characteristics is identified at various positions. Structure of spray deposit on basis of the nozzle velocity combination determines the strain effects [8].

3. Research methodology

In this section, Design of the optimal flow characteristics of the spray droplet and velocity distribution of UAV with respect to nozzle design on the particular environment and particular chemical has been carried out using turbulence model with following research objective. Further computational fluid dynamics is carried out using solid works and CATIA [9]. Its design is evaluated as simulation using the ANSYS

3.1. Design Objectives of the Spray Quality of the Plant Protection UAV

- It is to reduce the spray drift with respect to the structure of spray deposit
- It is to increase optimal flow characteristics of droplet with respect to speed and size
- It is to increase the volume distribution of droplet
- It is to provide optimal velocity distribution of the UAV
- It increases the uniformity of the liquid distributions on controlling the various spray effects
- It is to identify the optimal nozzle orientation and nozzle distancing for uniform distributions
- It is to analyze the biological effectiveness of the applied chemical and environment risk on basis of the velocity and size of the droplet.
- It is to optimize Cone Straight Nozzle on Nozzle Geometry using Controlled variable Method

3.2. Design of the Spray Nozzle of UAV using Computational Fluid Dynamics model through Solid Works

Design of the spray nozzle of unmanned Aerial Vehicle using computational fluid dynamics model is designed using various optimal flow characteristics and velocity distributions to meet above mentioned objectives. Design is carried out using CATIA software on modeling the design specification tree. Sprayer nozzle is important component in the spraying operations towards uniform liquid distributions. Nozzle is fabricated using various materials such as aluminum, copper, Brass, Stainless Steel and ceramics. It is available in shapes such as flat fan, cone type, hollow cone and straight cone [10].

Optimal Parameter to the spray quality of the UAV determine the model efficiency to with respect to the position and distribution of the droplet. Further nozzle design of the spray is optimized reduce the spray drift and increase the volume distribution on basis of the droplet size and velocity of the droplet. Further losses during the spraying leads to effective spraying design based on different factors. Especially droplet size and target drift are important factor for efficient of spraying system. Further Characteristics of the sprayer nozzles with respect the structure of spray deposit effect the velocity and size of the droplet

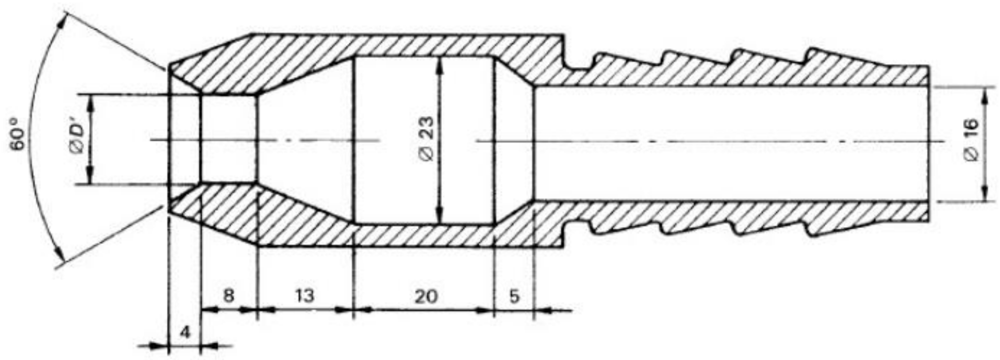
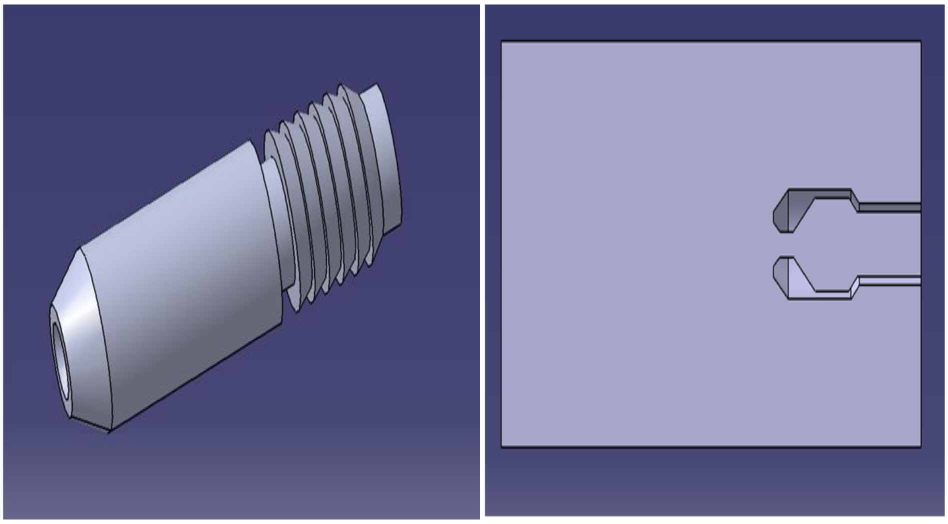


Figure 1: Nozzle Geometry

In addition, Specification of Tree structuring along the appropriate features for producing the optimal flow characteristic of drop let and velocity distribution of the UAV on using the straight cone nozzle. Cone nozzle has better atomizing ability by comprising converging section, throat section and reaming section [11]. The converging section is used to concentrate the fluid and energy, the throat section to stabilize the flow status, and the reaming section to control jet exit diffusion angle and control cavitation.Influence of the nozzle geometry with respect to converging angle and throat length is simulated to identify the optimal distributions.

3.2.1. Design of Cone Straight Nozzle optimization on Nozzle Geometry using Controlled variable Method

The nozzle geometry contains two primary sections termed as the contraction section and the focus section is represented using computation fluid dynamics 2D model. Model uses the controlled variable method on the boundary condition of the various section.In contraction section, nozzle cross-sectional area is represented with pressure of 250Mpa and density with 2600kg/m, the velocity of the mixture represented as mass flow rate as 0.1kg/s, and the abrasive particles are accelerated with diameter of 0.18mm. In Focus Section, velocity of the mixture reaches the peakwith pressure of 0.6Mpa. Figure 2represents the nozzle geometry of the spray unit of the UAV with solid surface.

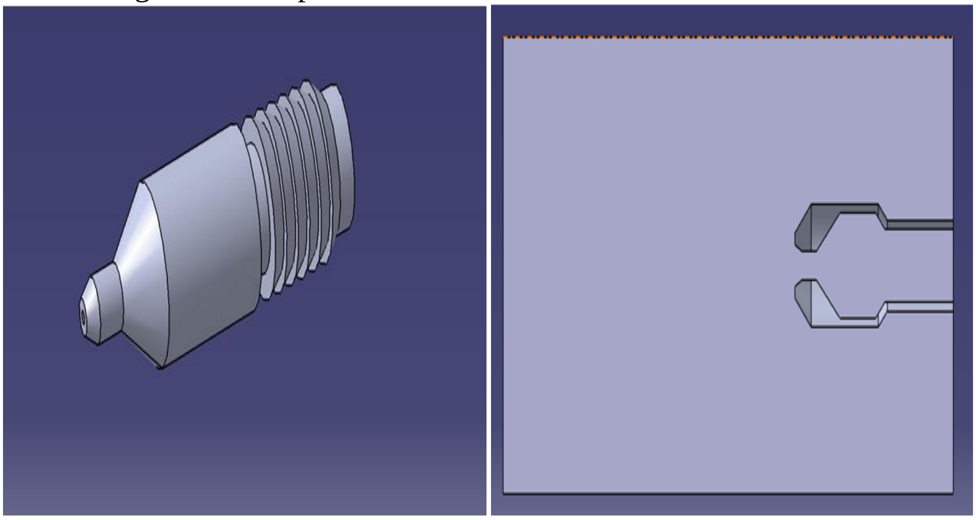


(a) **(b)**

Figure 2: Cone Nozzle Geometry (a) Cone Nozzle (b) Solid Surface of Cone Nozzle

4. Simulation Analysis

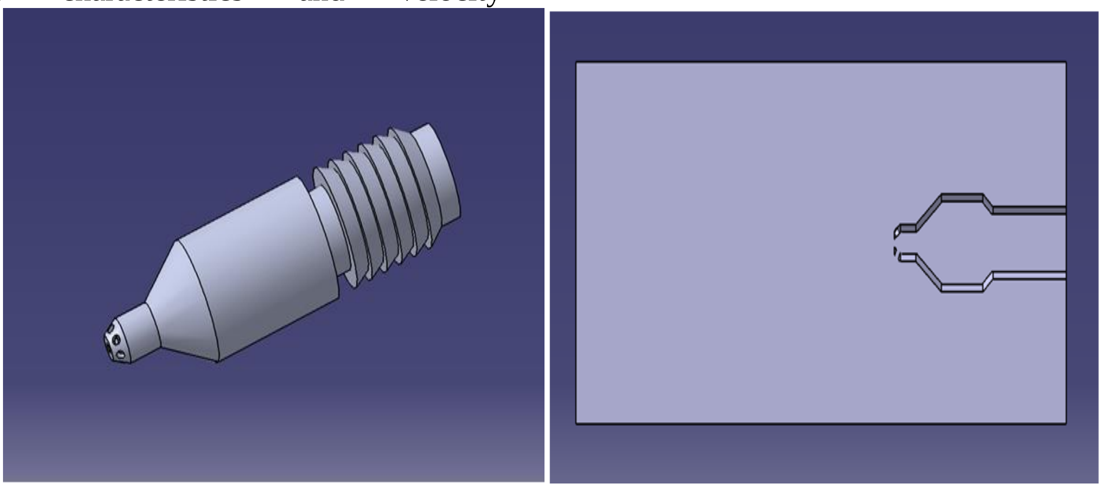
In this section,Analysis of Computation Fluid Dynamics on reduced angle of the cone nozzle with multiple holes is carried out using ANSYS software with respect to increasing the spray speed and spray volume to target. Analysis includes selecting the optimalflow characteristics and velocity distribution of the nozzle to evaluate the efficiency of the spray unit of UAV to target on various biological and environment condition of target. Evaluation of the nozzle geometry on basis of angle reduction is represented in the figure 2



(a) **(b)**

Figure 3: Illustration of the Angle Reduced Cone (a) Angle Reduced Cone and (b) Solid Surface

Evaluation of the multiple hole structure of the cone Nozzle employed to the pesticide spraying to the target on the flow characteristics and velocity distribution using controlled variable method along boundary condition is represented in the figure 4

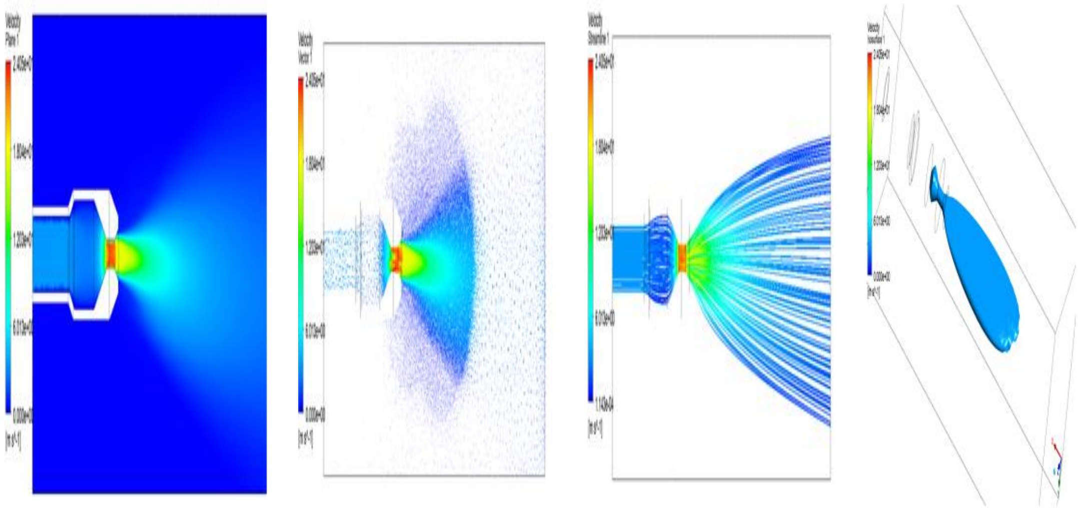


(a) **(b)**

Figure 3: Illustration of the Angle Reduced Cone with multiple holes (a) Angle Reduced Cone and (b) Solid Surface

Computation analysis

Computational Analysis is carried out in ANSYS CFX software to measure the spray volume and spray speed to cone straight nozzle to the spraying dose to the target with respect to the target boundary condition. ANSYS CFX is fluid dynamic program with meshing technologies to provide computational fluid dynamics solutions for fluid structure interaction problems [12]. The CFD analysis is based on the energy equation, momentum equation and continuity equation for droplet size and its volume distribution to the target.



(a) (b) (c) (d)

Figure 4: Illustration of the Cone Nozzle (a) Normal Velocity of Fluid (b) Vector Velocity of Fluid (c) streamline Velocity of the Fluid (d) ISO Surface Velocity of the Fluid

Evaluation of the computational fluid dynamics to the cone straight nozzle to the spraying dose on the spray drift, spray angle and spray speed and spray volumes on boundary conditions such as velocity and droplet rate is represented in figure 4 with normal velocity, vector velocity, ISO surface and streamlined velocity. Initially Pressure conditions [13], density and velocity of the particle is fixed to the applications. Figure 5 represents the velocity flow of the fluid with respect to the volume on the cone nozzle of the application. The water flow from nozzle which having mass flow rate1.037kg/s through the domain volume.

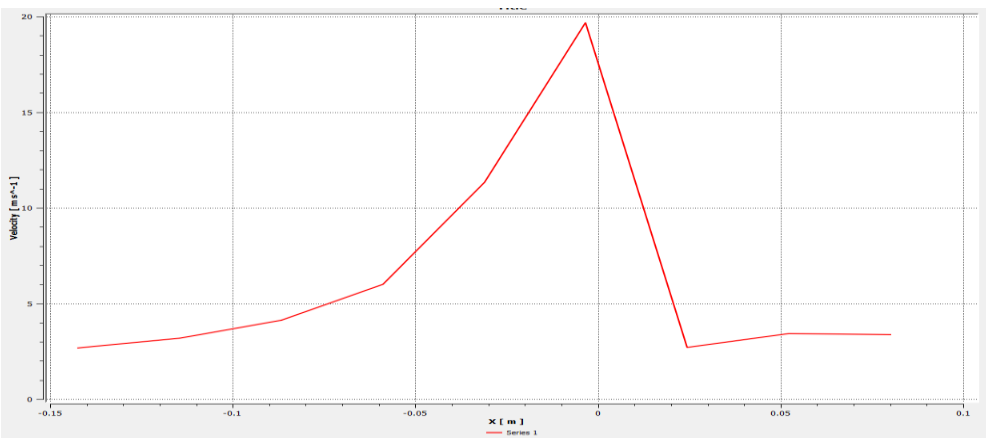
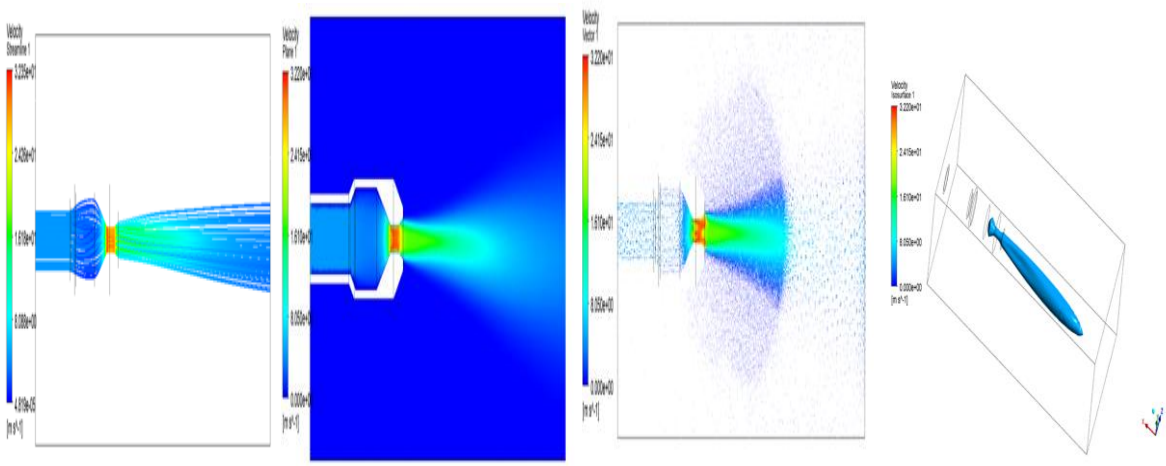


Figure 5: Evaluation of the Cone Nozzle Velocity against the volume

Computational Analysis is carried out in ANSYS CFX software to measure the spray volume and spray speed to the multiple holes of angle reducecone straight nozzle to the spraying dose to the target with respect to the target boundary condition. ANSYS CFX is fluid dynamic program with meshing technologies to provide computational fluid dynamics solutions for fluid structure interaction problems. The CFD analysis [14] is based on the energy equation, momentum equation and continuity equation for droplet size and its volume distribution to the target.



(a) (b) (c) (d)

Figure 6: Illustration of the angle reduced Cone Nozzle (a) Plain Velocity of Fluid (b) Vector Velocity of Fluid (c) streamline Velocity of the Fluid (d) ISO Surface Velocity of the Fluid

Evaluation of the computational fluid dynamics of angle reduced cone straight nozzle to the spraying dose achieves high efficiency in reducing the spray drift, spray angle and increasing the spray speed and spray volumes on boundary conditions such as velocity, and droplet rate is represented in figure 6 with normal velocity, vector velocity, ISO Surface Velocity and streamlined velocity. Initially Pressure conditions, density and velocity of the particle is fixed to the applications. Figure 7 represents the Evaluation of the Angle Reduced Cone Nozzle Velocity against the volume.The water flow from nozzle which having mass flow rate 1.247kg/s through the domain volume.

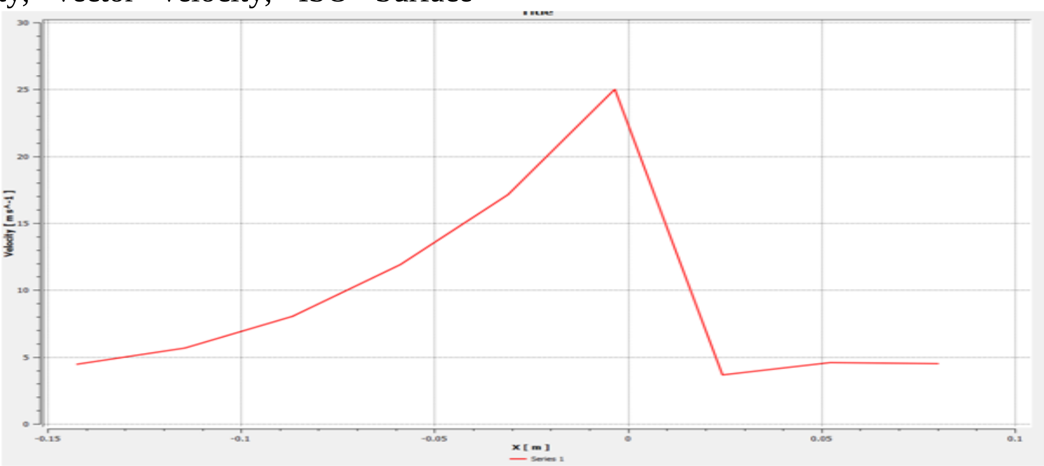
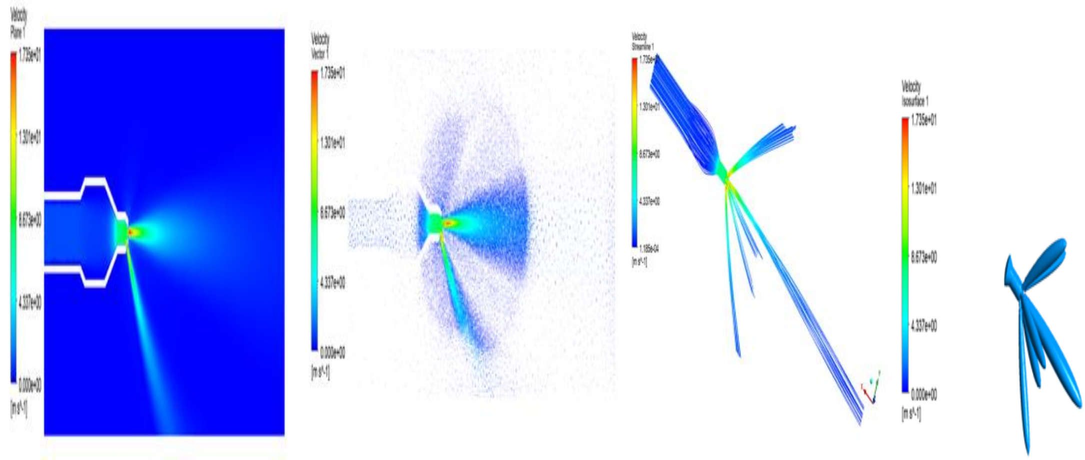


Figure 7: Evaluation of the Angle Reduced Cone Nozzle Velocity against the volume

Computational Analysis is carried out in ANSYS CFX software [15] to measure the spray volume and spray speed to the multiple holes of angle reducecone straight nozzle to the spraying dose to the target with respect to the target boundary condition. ANSYS CFX is fluid dynamic program with meshing technologies to provide computational fluid dynamics solutions for fluid structure interaction problems. The CFD analysis is based on the energy equation, momentum equation and continuity equation for droplet size and its volume distribution to the target.



(a) (b) (c) (d)

Figure 8: Illustration of the Cone Nozzle with multiple hole (a) Normal Velocity of Fluid (b) Vector Velocity of Fluid (c) streamline Velocity of the Fluid (d) ISO Surface Velocity of the Fluid

Evaluation of the computational fluid dynamics to the multiple hole of angle reduced cone straight nozzle to the spraying dose achieves high efficiency in reducing the spray drift, spray angle and increasing the spray speed and spray volumes on boundary conditions such as velocity, and droplet rate is represented in figure 8 with normal velocity, vector velocity, ISO Surface Velocity and streamlined velocity. Initially Pressure conditions, density and velocity of the particle is fixed to the applications. Figure 9 represents the Evaluation of the Angle Reduced Cone Nozzle with multiple holes on Velocity against the volume.The water flow from nozzle which having mass flow rate 1.247kg/s through the domain volume.

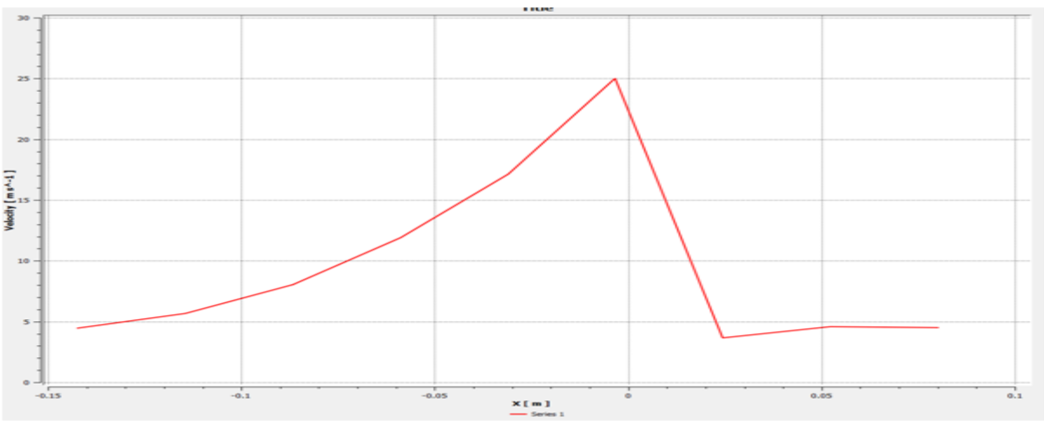


Figure 9: Evaluation of the Angle Reduced Cone Nozzle with multiple holes Velocity against the volume

Finally, cone nozzle is evaluated on the computational fluid dynamic model is represented in the table 1 against the velocity and volume with respect to mass flow rate of the fluid to the target medium on the boundary conditions to the turbulence model along the various setting of the fluid medium with respect to the nozzle design such as angle reduced cone nozzle, angle reduced multiple hole cone nozzle for fluid displacement.

Table 2: ANSYS Analysis of Computational fluid dynamics of spray on cone nozzle

Cone Nozzle	**Plane Velocity**	**Streamline Velocity**	**Vector Velocity**	**Iso Surface Velocity**	**Mass Flow Rate**
Normal reduced Cone Nozzle	**2.405e+0ms**	**2.605e+0.1ms**	**2.705e+0.1ms**	**2.805e+0.1ms**	**1.11kg/s**
Angle Reduced Cone Nozzle	**3.235e+01ms**	**3.22e+01ms**	**3.24e+01ms**	**3.28e+01ms**	**1.037Kg/S**
Angle Reduced Cone Nozzle with Multiple holes	**1.74e+01ms**	**1.78e+01ms**	**1.73e+01ms**	**1.735e+01ms**	**1.247Kg/s**

Conclusion

In this paper, design of the angle reduced cone nozzle is carried out using various fluid dynamic velocities. Angle reduced cone nozzle employed for spray unit of the agriculture UAV is optimized with process parameter for improved turbulence flow and spray quality with reduced spray drift. Computational fluid dynamic is carried out on ANSYS software to evaluate the spray speed, spray volume of cone nozzle with and without angle reduction to the spray unit to produce the continuous distribution of the dose to the target crop towards pesticide applications. Based on the analysis, it is concluded that optimal parameter to cone nozzle is considered to produce better velocity and speed of the fluid with mass flow rate of 1.247Kg/s.

References

1. Aditya S. Natu., Kulkarni, S., C. (2016) "Adoption and Utilization of Drones for Advanced Precision Farming: A Review." published in International Journal on Recent and Innovation Trends in Computing and Communication, ISSN: 2321-8169, Volume
2. Clijmans, L., Swevers, J., De Baerdemaeker, J., & Ramon, H. (2000). Sprayer boom motion, part 1: derivation of the mathematical model using experimental system identification theory. Journal of Agricultural Engineering Research, 76(1), 61-69
3. Costa, F., Ueyama, J., Braun T, Pessin G, Osorio F, Vargas P. (2012) "The use of unmanned aerial vehicles and wireless sensor network in agricultural applications.", IEEE conference on Geoscience and Remote Sensing

Symposium (IGARSS-2012) 5045–5048

4. F. G. Costa, J. Ueyama, T. Braun, G. Pessin, F. S. Osorio, P. A. Vargas, "The Use of Unmanned Aerial Vehicles and Wireless Sensor Network in Agriculture Applications,
5. Holownicki, R., G. Druchowski, Swiechowski, W. and Jaeken, P. (2002). Methods of evaluation of spray deposit and coverage on artificial targets. Electronic Journal of Polish Agricultural Universities, Agricultural Engineering, 5(1), 129-136
6. Issue: 5 PP.563 - 565. 4. Giles, D. K., & Billing, R. C. (2015) "Deployment and Performance of a UAV for Crop Spraying. Chemical Engineering Transactions, 44, pp.307- 322
7. Kulkarni, S., C. (2016) "Adoption and Utilization of Drones for Advanced Precision Farming: A Review." published in International Journal on Recent and Innovation Trends in Computing and Communication, ISSN: 2321-8169, Volume
8. Komal Tanaji, Nimbalkar Aishwarya Satish, "Agriculture Drone for Spraying Fertilizers and Pesticides", Sept 2017, International Journal of Research Trends and Innovation, ISSN 2456-3315, Volume 2, Issue 6
9. Prof. B.Balaji, Sai KowshikChennupati, Siva Radha Krishna Chilakalapudi, Rakesh Katuri, kowshikMareedu, "Design of UAV (Drone) for Crops, Weather Monitoring and For Spraying Fertilizers and Pesticides.", Dec 2018, IJRTI, ISSN: 2456-3315
10. Prof. P. Mone, Chavhan Priyanka Shivaji, Jagtap Komal Tanaji, Nimbalkar Aishwarya Satish, "Agriculture Drone for Spraying Fertilizers and Pesticides", Sept 2017, International Journal of Research Trends and Innovation, ISSN 2456-3315, Volume 2, Issue 6
11. Qin W C, Xue X Y, Zhou L X, Zhang S C, Sun Z, Kong W, et al. Effects of spraying parameters of unmanned aerial vehicle on droplets deposition distribution of maize canopies. Transactions of the CSAE, 2014; 30(5): 50–56
12. Qiu B J, Wang L W, Cai D L, Wu J H, Ding G Y, Guan X P. Effects of flight altitude and speed of unmanned helicopter on spray deposition uniform. Transactions of the CSAE, 2013; 29(24): 25–32
13. Rahul Desale, Ashwin Chougule, Mahesh Choudhari, Vikrant Borhade, S.N. Teli, "Unmanned Aerial Vehicle for Pesticides Spraying" April 2019, IJSART, ISSN: 2395-1052
14. S.R. Kurkute, B.D. Deore, Payal Kasar, Megha Bhamare, Mayuri Sahane, "Drones for Smart Agriculture: A Technical Report", April 2018, IJRET, ISSN: 2321-9653
15. St. Joseph, Mich.: ASAE. Ferguson, J. C., Chechetto, R. G., Hewitt, A. J., Chauhan, B. S., Adkins, S. W., Kruger, G. R. and O'Donnell, C. C. (2016). Assessing the deposition and canopy penetration of nozzles with different spray qualities in an oat (Avena sativa L.) canopy. Crop Protection, 8(1), 14-19